

RotCFD Software Validation - Computational and Experimental Data Comparison.

Ovidio Montalvo Fernández



NASA Ames Research Center – Aeromechanics Branch,
Universidad Autónoma de Nuevo León (UANL).
ovidiomontalvo@hotmail.com.

Abstract: RotCFD is a software intended to ease the design of NextGen rotorcraft. Since RotCFD is a new software still in the development process, the results need to be validated to determine the software's accuracy. The purpose of the present document is to explain one of the approaches to accomplish that goal.

Keywords: CFD, CAD Modeling, Rotorcraft, NextGen aircraft.

I. ACKNOWLEDGEMENTS

For all involved in this wonderful experience, and for all the knowledge acquired during this amazing time, I give the main credit to God who has blessed me with the opportunity of being where I am and allowed me to enrich my life during all this time; I know it had never been possible without you.

I owe special thanks to my family for all their unconditional support in everything I needed and for being there with me without hesitation helping me in every possible way.

I'm deeply grateful with my beloved partner Ana Lucía for being the reason to force myself to persevere, and for being there any moment, anywhere, anyhow. Thanks for all your love, dedication and great patience. All my successes are dedicated to you and my parents.

The success and final outcome of this project required a lot of guidance and assistance from many people and I am extremely fortunate to have got this all along the completion of my project work. Whatever I have done is only due to such guidance and assistance and I would not forget to thank them. I especially want to thank Dr. William Warmbrodt for giving me the opportunity to be at NASA Ames Research Center working in the Aeromechanics Branch and for have trusted in my skills and knowledge to perform as a member of such an important team.

I am thankful to and fortunate enough to get constant encouragement, support and guidance from Eduardo Solis

who was always looking for our welfare and for us to have the best work performance in our own benefit.

I would not forget to remember - for their unlisted encouragement and more over for their timely support and guidance to Shirley M. Burek, Meridith Segall, Guillermo Costa, Ashley Pete, and Carl Russell.

I owe a great many thanks to many people who helped and supported me during the process preceding my internship at NASA as well as those who were tracking the whole process; thanks to my institution and my faculty members without whom this project would have been a distant reality; thank you to all you!

II. NOMENCLATURE

VTOL – Vertical Take-off and Landing.

CFD – Computational Fluid Dynamics.

Downwash – Air forced downward as a result of the momentum provided by an airplane wing or a rotor blade.

Outwash – Air forced outward in a rotor.

ρ – Fluid density

\vec{V} – Velocity field.

\vec{f} – Body force.

II_{ij} – Stress tensor.

\vec{S} – All terms not accounted in the equation i.e. Sources.

III. INTRODUCTION

NASA is aware of the high and constantly growing aircraft congestion in the biggest airports all around the world, this is why it has started to design new aircrafts which can easily be accommodated by current and future airports. One of the presented solutions is the design of VTOL civil aircraft. The

main advantage of this designs is that they don't necessarily need a long runway for takeoff or landing.

A great deal of the new designs are aircraft that use rotors in some way as a mean of lift or propulsion generation, and that is why an efficient and easy-to-use CFD tool is needed to generate all the analyses needed for the design.

Sukra Helitek Inc. and its newest software – RotCFD – were selected to work in conjunction with NASA in the development of this software, which will make the design of NextGen rotorcraft much easier [1].

IV. THEORETICAL FRAMEWORK

III.1 Computational Fluid Dynamic Fundamentals

Computational Fluid Dynamics, also known by the acronym CFD, is the branch of the fluid dynamics used to predict fluid flow behaviors involving heat transfer, chemical reactions, viscosity, etc. by means of computer-based simulations.

All of CFD in one form or another is based in the fundamental governing equations—of fluid dynamics: The continuity, momentum, and energy equations which in turn, are the mathematical representation of the fundamental physical principles: mass conservation, momentum conservation (Newton's second law) and energy conservation (first law of thermodynamics).

Having the fundamental physical principles is necessary to apply them to a suitable model of flow; among these models are the fixed finite control volume, moving finite volume, fixed infinitesimally small volume or moving infinitesimally small volume in order to obtain the mathematical equations which embodies the aforementioned physical principles.

The fluid flow considered in CFD is assumed to be a continuum medium in which the Navier-Stokes equations can be applied; however these are nonlinear partial differential equations and their solution is very difficult. At this time, the computational resources are not capable enough to handle such complex problems (at least in the commercial use). This is the reason why the Navier-Stokes are usually simplified, neglecting some terms.

The flow model used on RotCFD is the one in which an infinitesimally small element of fluid is fixed in space having a differential volume dV . The fluid is infinitesimal in the same sense as differential calculus; however, it is large enough to contain a huge number of molecules so that it can be viewed as a continuous medium. Instead of applying the fundamental principles to the whole flow they are applied to the infinitesimally small fluid element itself [2].

The fluid flow in the present paper is governed by the unsteady, laminar, incompressible Navier-Stokes equations. For inviscid incompressible flow, conservation of mass and momentum are sufficient conditions for solving the flow field, thus making the energy equation redundant.

The conservation of mass law applied to a fluid passing through the infinitesimally small fixed control volume aforementioned yields to the following equation of continuity:

$$\frac{\partial}{\partial t}(\rho) + \nabla \cdot (\rho \vec{V}) = 0$$

Newton's second law applied to a fluid passing through an infinitesimal control volume could be expressed by the momentum equation as shown below:

$$\frac{\partial}{\partial t}(\rho \vec{V}) + \nabla(\rho \vec{V} \vec{V}) = \rho \vec{f} + \nabla \vec{\tau} + \vec{S}$$

III.2 Discretization

The aforementioned equations assume that all dependent variables change continuously through the domain. If the equations are applied to a large domain, the variations in midpoints would be unknown or inexact. This is the reason why it is necessary to "break the domain up" into small sections and apply the governing equations in those small parts instead; in this way, an accurate estimation of the flow behavior in all the domains can be obtained. This process is known as discretization.

Formally defined, discretization is the process by which a closed-form mathematical expression, such as a function or a differential or integral equation involving functions is approximated by analogous expressions which prescribe values at only a finite number of discrete points or volumes in the domain.

In contrast to an analytical solution of partial differential equations, in which the variation of dependent variables are given continuously throughout the domain, the numerical solution can give answers only at discrete points in the domain, called grid points.

The nature of the resulting algebraic system depends on the character of the problem posed by the original partial differential equation. Equilibrium problems usually result in a system of algebraic equations that must be solved simultaneously throughout the domain in conjunction with specified boundary values. [3].

There are different kinds of discretization methods employed in the CFD field; among the most common are the finite difference, finite element, and finite volume. This last one is the method employed in RotCFD.

One of the advantages of the finite volume approach resides in the easy grid adaptability to the body surface and the flexibility for the transition to the boundaries. Computationally speaking, the finite volume approach is able to work in a "physical" plane without the need of a transformation from physical to a computational plane just as finite differences approach does. The elimination of this step allows the use of complex shaped grid elements [3].

III.3 Momentum source approach

The momentum source approach, first applied to vertical axis wind turbines, simulates the rotor effect on the flow by means of the momentum generated by the rotor blades' geometry. The rotor is replaced by distributed sources of momentum in the flow. The direction and magnitude of the moment depends on the rotor geometry and the local flow characteristics. The main advantage of this approach is that it doesn't need a body-fitted rotor grid that requires a lot of computational resources [4].

V. METHODOLOGY

The way chosen for the validation process is based in the comparison between experimental and computational results obtained for some rotorcraft; among these are the CH-53, CH-47 Chinook, UH-60 BlackHawk, and the V-22 Osprey. Specifically, the results will be compared using outwash & downwash analyses for the previously mentioned rotorcraft [5].

The first step was the CAD modeling of the four aircraft. The technical drawings were obtained from the web and they are of public domain.

The software used for the CAD modeling was CreoParametric2®. Front, side and top technical views were placed in the three computational planes accordingly to their location one respect to the others (Fig. 1). Special attention has to be placed in the alignment and the dimensioning at this step since any misalignment can create complications later on the modeling (i.e. 3d curves may not intersect).

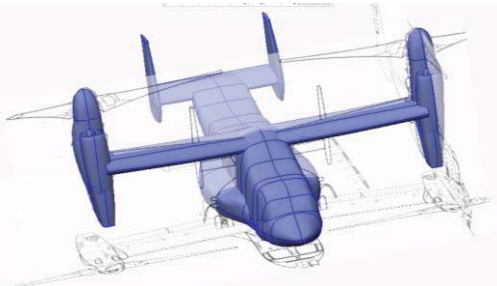


Fig. 1 3D Modeling - CreoParametric2.

Once the model was finished in CreoParametric2, it was exported as an IGES file, which allowed the model to be imported in a second CAD software called Rhinoceros®. The next step, called post-modeling, consisted of a series of procedures in which the geometry was treated in order to prepare it for RotCFD. The model has to be examined meticulously; if any hole is found, even a tiny one, it has to be patched. All the surfaces have to be joined together, creating a kind of single-surfaced body. If all the previous steps are made exactly as stated before, a lot of troubles will be avoided on the grid generation. The final four aircraft models are shown on fig. 2.

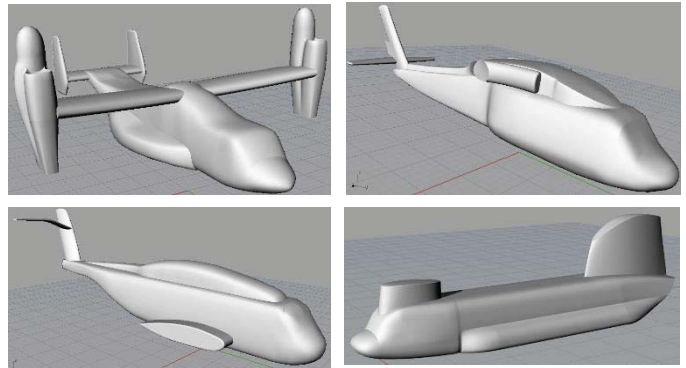


Fig. 2 3D Models - Rhinoceros (V-22, UH-60, CH-53, Ch-47).

RotCFD is capable of outputting two kind of files: STL and P3D; therefore there are two different ways to follow before RotCFD.

1. Export the geometry from Rhinoceros as an STL file and then import it into the module RotCFD UNS.
2. Export the geometry from Rhinoceros as an STL file, import it then into the RotCFD module called ShapeGen, save it as a P3D file and finally import it into RotCFD UNS.

The first option is the fastest one but the file size can be too large. The second option is recommended since it has just an extra and simple step and yields a more manageable file. The extra step does not require a special treatment of the geometry it is as simple as import the file and save it as a P3D file.

One important thing about the STL creation from Rhinoceros is the surface mesh quality: the mean aspect ratio of the triangles which conform the surface has to be kept small. Their size also needs to be small enough to generate a smooth surface instead of sharpened faces.

The following figure is an example of the geometry generated using inappropriate mesh sizes. A well-parameterized mesh would look at first glance as the original geometry.

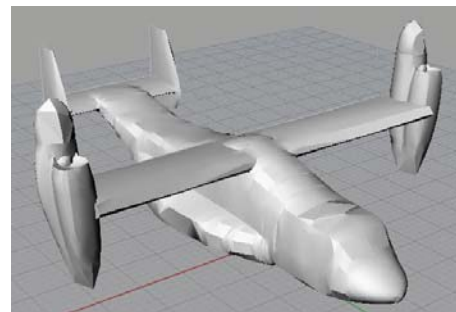


Fig. 3 Unacceptable STL model.

The format manipulation process using different software prior to RotCFD can be synthesized in the diagram shown on Fig. 4.

Once the model has been imported into RotCFD UNS, the location of the aircraft is simple and intuitive. It can be located accordingly to global coordinates or using relative frameworks; rotors or others aircraft can be added. RotCFD also allows the user to change the pitch, roll and yaw relative to the global coordinate axes (default: Pitch: X rotation, Roll: Y rotation, Yaw: Z rotation).

The rotor generation is a very simple step on RotCFD and is what makes it different from other CFD software. In a conventional CFD software it is necessary to model the rotor blade geometry and create a body-fitted mesh attached to the rotating blade. This kind of mesh makes the calculations much more difficult, and greatly increases the time required to generate the grid and the solution.

RotCFD, on the other hand, calculates the flow through the rotors using the *momentum source* approach, which simulates the rotor influences on the flow by applying a series of momentums equivalent to the ones generated by the original rotor. There are a certain number of “*sources*” placed along the radius of the rotor which represent the momentum imparted to the flow. The momentum magnitude and direction depends of the flow characteristics and the airfoils sections of the blade.

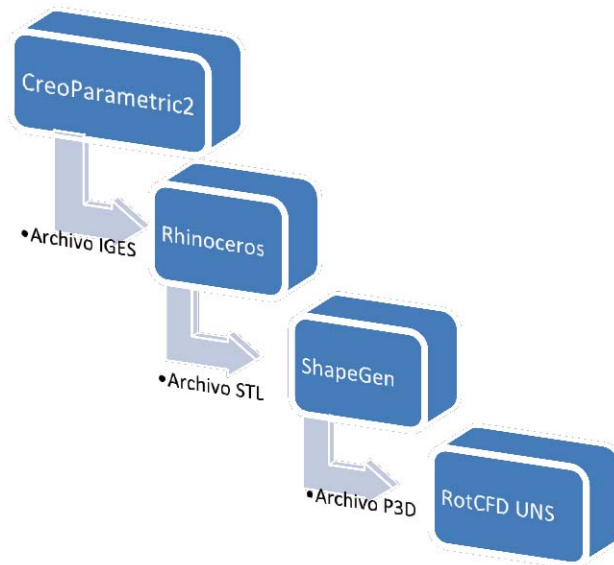


Fig. 4 Process prior to RotCFD.

The rotor location is basically the same as the aircraft body previously mentioned. The inputs for the rotor are the follows:

- Rotor radius: Total rotor radius.
- Cone angle: The angle between the blades respect to the hub horizontal axis.
- Collective pitch: The pitch angle applied to all the blades at the same time.
- Reference twist: Twist angle at 75% of the rotor radius.
- Tip velocity: Rotor tip velocity.
- Cutout radius: ratio between the physical start of the blade and the total rotor radius.

- Hinge radius: Ratio between actual hinge length and rotor radius.
- Rotation direction: Clockwise or counter-clockwise.
- Source locations: Number of momentum sources along the rotor radius.
- Cyclic pitch: Cyclic pitch coefficients.
- Flapping: Harmonics flap coefficients.
- Airfoil tables: Selects the airfoil used at different fractions of the rotor radius.
- Chord/Radius: Chord/radius ratio at different fractions of the rotor radius.
- Twist: Twist angles at different fractions of the rotor radius.
- Outplane deflection: It allows the user to specify the out of plane deflection at different fractions of the rotor radius.

The next step once the model has been imported, located, and the rotor has been created is to generate the boundaries that will determine the analysis area. The distances and specifications for each side of the walls can vary depending on the kind of analysis. Specifically for this case in which the helicopter is in hover flight far away from ground or any other object, the boundaries are located at distances where the air is barely influenced by the rotor and the body. In the case in which the flow does interact with real walls (as the ground and wind tunnel test section walls are) the boundary walls are located at the same location that the real ones (fig. 5). The way to make them “real” is by specifying conditions of impenetrability, this is, make the normal component of the velocity equal to zero.



Fig. 5 Boundaries and refinement boxes.

One of the most critical steps is the grid generation. The greater the interest of the flow behavior in certain areas, the smaller the grid size has to be. The same applies for high pressure gradients or for complex geometries where high definition is needed.

Some of the most important factors to take into account specifically on RotCFD are the follows:

- Initial X (Y or Z) Cells: This number defines the number of divisions of the total length between two boundaries placed perpendicular to the direction specified (X, Y, or Z). They are called “initial” number of cells because if there is an object with a specific refinement somewhere in the middle or near enough to the trajectory, those cells will be divided in even smaller elements (depending on the body refinement).
- It is important to mention that the ratio total length/number of divisions, has to be kept constant

for all three directions; in other words, each of the elements formed in the grid has to be a perfect cube. The mathematical behavior of the governing equations is very complicated, and if this ratio is maintained, the computational solution will be much easier to obtain.

- **Body Refinement:** Allows the grid to be smaller in the surrounding of the body. 6 or 7 are common values used.
- **Rotor Refinement:** Allow the grid to be smaller in the rotor surroundings. 6 or 7 are common values used.
- **Fit bodies by default:** This box enables an algorithm that makes the rectangular grid adapt to the body creating a tetrahedral grid near the body. If the box is not checked the geometry will be simplified too much and at the end it can give results different to the expected ones.

Besides all these results there is another tool that increases the grid refinement in specified areas, this is called *Refinement Box*. As its name implies this tool generates a simple box that increases the refinement of the grid in all the volume contained on it. Usually they are used in zones of special interest, i.e. rotor surroundings, body surroundings, on the ground for Outwash & Downwash analyses (fig. 6).

We have shown up to this point all the steps and the inputs needed for the grid generation.

Once all is ready the grid is generated by pushing the “Run Grid Generator” button. The results looks like the follow screenshot.

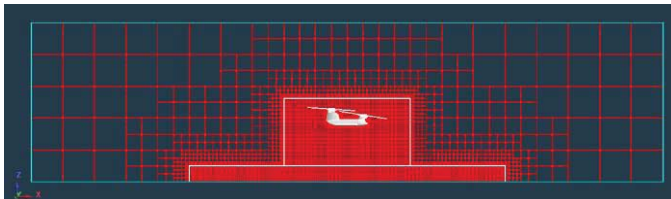


Fig. 6 Grid.

In some cases the geometry have small holes, in such cases the grid would be generated even into the body. One way to verify that everything is working well up to the present step is by hiding the body as shown in fig. 7. In this specific case the grid was generated properly what means that the geometry was well-modeled.

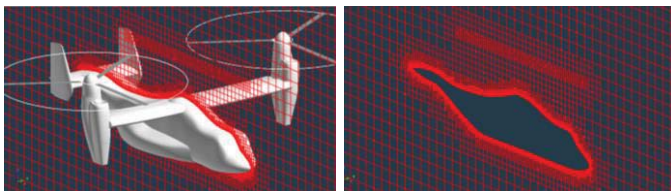


Fig. 7 Grid inspection.

RotCFD has a helpful tool called “Body Surface Tester” that enables the visualization of the geometry that actually is going

to be analyzed (The original geometry is just a medium for the last one).

As previously mentioned, the definition of the final body depends on the grid but also on the original geometry.

In figure 8 two different V-22 geometries are shown, the only difference between both is the grid refinement. In the first one the geometry looks almost as the original geometry, the only differences are in thin parts such as the vertical stabilizer or the wing trailing edge. Moreover the fig. 8 (b) looks a little coarser with a surface not as smooth as the previous one.

The geometry refinement is an important factor buy it is not the only one to take into account; reduction of time is something desirable on CFD analyses and an analysis using a grid as refined as the one used in fig.8 (a) will take much more time than the second one and sometimes the results are almost the same. This is why there is not just a single right solution. Trade-off between this two factors is essential in CFD.

The physical flow properties in RotCFD are essentially the same that the ones at any CFD software and all of them depend on the atmospheric conditions at which the analysis is taking place i.e. density, temperature, gas constant, specific heats, viscosity, pressure, etc. It is assumed that the reader knows all this properties therefore extra explanation is not included.

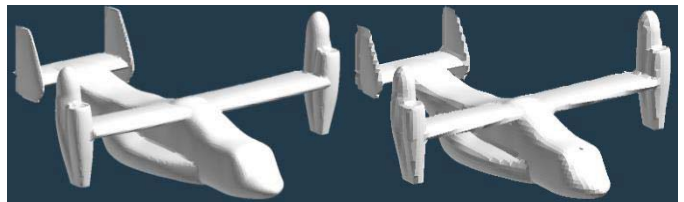


Fig. 8 (a,b) Body Surfaces comparison.

One of the tools RotCFD has specialized for aircraft is the “Flight Condition” module. This section makes the characterization of the flow direction and velocity very simple with minimal user interference needed. Five different flight conditions are available:

- **General:** It allows the user to specify the magnitude for all three velocity components.
- **Hover:** As this is a flight characteristic in which the velocity relative to the rotorcraft is zero not input is needed here.
- **Forward Flight:** Only the velocity for the direction of the flight is needed.
- **Climb:** Only the climb velocity is needed and it is not necessary to specify directions.
- **Descent:** Only the descent velocity is needed and it is not necessary to specify directions.

The specification of the grid times and iterations is the last step before running the simulation. The options available are:

- **Time Length:** This is the physical real time that the software is going to calculate, the analysis start calculating the flow at time zero with the boundary conditions and flow properties as initial conditions

i.e. Analyze the rotor wake the first 20 seconds once the rotor star to spin.

- **Time Step:** This is the number of divisions that the Time Length will be spitted for the calculus. The grid solution will be calculated at small time increments of $\Delta t = \text{Time Length} / \text{Time Step}$. The larger the Time Step, the larger the results are for each grid discrete point. The results are also more accurate but the simulation time will take longer.
- **Iterations:** Number of iterations for each Δt at each grid discrete point.
- **Relaxation numbers:** They prevent large increments in the equation dependent variables from one iteration to other. The value 0.01 means that only 1% of the present variable value is been used for the calculus the remaining 99% depends of the previous iteration.

Up to this point everything needed for the simulation is ready but is recommended to enables the “Save restart history” option because is very common to have the analysis stopped for any reason, in that case all the information previously calculated would be lost.

The simulation time can vary from hours up to weeks or months depending on computer capabilities and grid refinements. It is recommended to run some simple simulations at first using coarse grid refinement and verify that everything works as expected, so then increase refinements and run loner simulations.

VI. RESULTS

As was suggested, some preliminary analyses were run at first and the grid was inspected to verify that everything was working properly. Fig 9 shows some of these preliminary analyses.

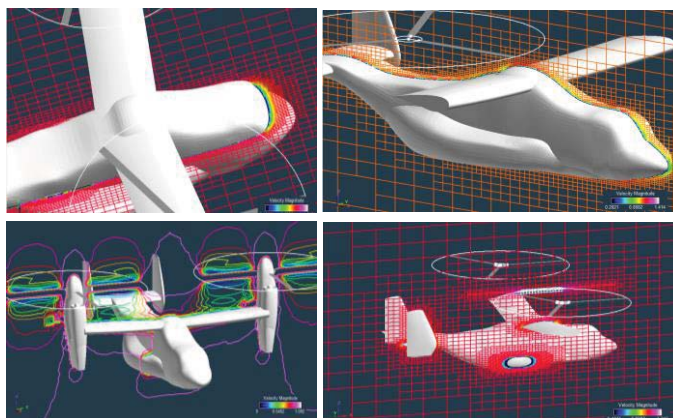


Fig. 9 Preliminary results

Having these results is possible to assume that the grid is properly working, unfortunately these were not the only results but also there were some results in which some errors

were presented giving some extremely high pressure gradients in very small areas.

In general CFD does not offer straightforward solutions that's why a lot of *Time Length*, *Time Step*, *Iterations* and *Relaxations number* analyses combinations were run in order to find the best combinations. Some of the most stable results were found using relaxation numbers of 0.01, Time Length higher to 10, Time Steps in a range from 1000 to 10000 and Iterations around 20.

In the fig. 10 are shown the screenshots in where is possible to see some of the different results using different combinations.

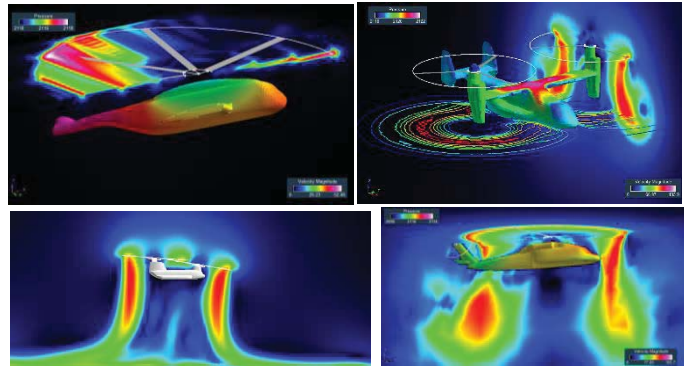


Fig. 10 Results

All the previous analyses were run at hover flight which means that the aircrafts are in a supposed single point without relative wind velocity.

In fig. 10 a) the CH-53 was analyzed at cero collective pitch, with a blade without twist and using generic airfoils NACA 0012. Basically the rotor is not generating lift force and this is why is not possible to see the wake under the rotor.

The V-22 analysis was run at 10 seconds with small increments of $\Delta t = 0.1$ and using collective pitch of 10° . In this analysis is possible to see how the wakes does develops under the rotor. The different colors on the surface are the pressure variations on the body surface due strictly to the flow. The highest pressure is presented in the upper wind which makes sense because is where the flow has the higher velocity normal to the body surface.

The helicopter CH-47 opposite to the others is analyzed near the ground and this is why the flow develops in the outward direction near the lowest boundary condition.

The last picture shows the UH-60 the one is using the same specifications than the V-22. Basically the difference is the interaction between the flow and the fuselage. In the V-22 the body pressure is higher in the upper part of the fuselage due to the high velocity flow under the tip of the rotors but in the UH-60 the highest velocities of the flow under the tip rotors are not above the upper part of the fuselage.

Essentially this is the whole procedure used for the validation of RotCFD results, due to data confidentiality, all the present analyses are not using the real rotor specifications neither the real airfoils. All resting is to change the above mention rotor characteristics, run the analyses and finally compare the experimental data whit RotCFD results.

VII. CONCLUSIONS

All the modeling processes and all previous procedure to the usage of a CFD tool have to be performed carefully and inspected over and over. Sometimes a simple error in the geometry can delay the project too much time if not detected on time.

The Project consummation will take place once all the rotor specification are changed for the same than the Outwash & Downwash ones and the analyses are run; having all these same specifications, both results are directly comparable.

VIII. REFERENCIAS

- [1] Sukra Helitek, Inc. "RotCFD Solver Theory" February 5 2013.
- [2] Anderson John Jr. "Fundamentals of Aerodynamics" 5th Edition 2011.
- [3] Anderson John Jr. "Computational Fluid Dynamics The basics With Applications" 1995.
- [4] Guntupalli Kanchan "Development, validation and verification of the Momentum Source Model for discrete rotor blades" owa State University 2011.
- [5] Smith Robert D. "Heliport/Vertiport Design Deliberations 1997-2000.

IX. ROJECT MEMBERS

William G. Warmbrodt¹, Carl R. Russell¹, José Luis Arciniega Jiménez², Zulema Guadalupe García Lozano², Erik Alberto Márquez Angeles², Eliud Israel Meza Escamilla³, José Alberto Ramírez Juárez²

¹ NASA Ames Research Center (NASA ARC).
Rotorcraft Aeromechanics Branch, ZIP. 94035.

² Universidad Politécnica Metropolitana de Hidalgo (UPMH).

Departamento de Ingeniería en Aeronáutica, CP. 42040.

³ Universidad Autónoma de Nuevo León (UANL).
Facultad de Ingeniería Mecánica y Eléctrica, CP. 66450.